BENCHMARKING OF THE ACTUAL INJECTION OF THE PRODUCT VERSUS PLASTIC SIMULATION SOFTWARE

ZUL AMRI BIN A.BAKAR

UNIVERSITI MALAYSIA PAHANG

SUPERVISOR'S DECLARATION

We hereby declare that we have checked this project and in our opinion this project is satisfactory in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering with Manufacturing

Signature : -----------------

Name of Supervisor :

Position :

Date :

Signature : -----------------

Name of Panel :

Position :

STUDENT'S DECLARATION

I hereby declare that the work in this thesis is my own except for quotations and summaries which have been duly acknowledged. The thesis has not been accepted for any degree and is not concurrently submitted for award of other degree.

Signature	:	------------------
Name	:	ZUL AMRI BIN A.BAKAR
ID Number	:	ME05015
Date	:	4 NOVEMBER 2008

DEDICATION

“Dedicated to my family the mission of my life”

ACKNOWLEDGEMENTS

Gratefully expressed from deep of my heart and highly gratitude dedicated toward my supervisor Mr. Zamzuri Hamedon for his precious guidance, invaluable ideas and knowledge, constantly encourage and support during easy and hard time to make this project available. Vast knowledge of him in multiple fields as well as technical and science conduction always impressing me and the feeling will persist to stay with me forever to be my model in achieving the future. I appreciate his consistent support from the first day of meeting until this concluding moment. I am truly grateful for his progressive vision about my training in science, his tolerances of my naive mistake, his patience in guiding my understanding, and his commitment to my future career. I also thankfully for the time spent for guiding me, transferring the knowledge and skills, and correcting my many mistakes and problems.

My sincere thanks to staffs of Mechanical Lab, UMP especially Mr. Khairul Azha in assisting, guiding and teaching me with his all dedications and sincere during all my days working at the lab. I am grateful for his patience in explaining my mistakes and misunderstanding which make this project available.

I am also indebted to my parents and family for their limitless love and supports during the progress of this project and as well as my whole life. I also dedicated this gratitude towards my colleagues for their ideas and support. Their encouragement and sacrifices make many unforgettable memories, knowledge and experiences for me during the years spent in UMP.

ABSTRACT

The idea of this project, benchmarking of the actual injection of the product versus plastics simulation software is to do analysis of the parameters that involved in the plastic injection molding in order to determine the best solution to overcome the problems and defects that occurred in the plastic injection molding. By actual injection processes, the results that can be observed are limited such as temperature, stress and the point of gate that are almost impossible to be analyzed by naked eye. Thus, for such details information the engineer can depend on Computer Aided Engineer (CAE) or Computer Aided Manufacturing (CAM) tools such as in this case Moldflow, in generating the accurate data of the parameter that has been analyzed. The tools are capable in assisting the designation of the mold and the parts that need to be produced by generates the data that cannot be achieved by doing the actual experiments. Hence, actual injection analysis needs quite a lot of effort to determine the optimal parameters for the injection process by experienced expertise. Try and error method was traditional way in injecting the part into fine product which consumed a lot of time and energy as well as increase the production cost. The result between software simulation and the actual injection might have slight differences because of several factors. The factors can be determined by doing both analyses and comparing the result will generated the data of error percentage of the simulation software to actual injection as same as factors of the errors.

ABSTRAK

Idea mengenai projek ini iaitu perbandingan antara injeksi sebenar produk dan perisian simulasi plastik adalah untuk menganalisa faktor-faktor yang boleh diukur yang terlibat secara langsung dalam arena acuan injeksi plastik. Ini adalah bertujuan untuk mencari jalan penyelesaian terbaik untuk mengatasi masalah dan kecacatan yang terdapat pada model injeksi plastik. Berdasarkan injeksi sebenar hasil analisis yang boleh diperhatikan adalah terhad kepada beberapa permerhatian sahaja dan faktor seperti suhu, tekanan dan titik kedudukan get adalah menghampiri mustahil untuk diperhatikan dengan menggunakan deria penglihatan manusia. Oleh itu untuk maklumat terperinci seperti perkara tersebut jurutera-jurutera boleh menggunakan perisian "*Computer Aided Engineer"* (CAE) atau "*Computer Aided Manufacturing"* (CAM) seperti dalam kes ini iaitu Moldflow untuk menghasilkan maklumat dan data yang tepat setelah menganalisa permerhatian tersebut. Perisian tersebut berupaya untuk menbantu dalam mereka cipta acuan dan model produk yang perlu dihasilkan dengan menghasilkan data yang tidak dapat diperolehi dengan melakukan eksperimen injeksi sebenar. Eksperimen injeksi sebenar memerlukan kepakaran dan tenaga yang tinggi untuk menghasilkan keadaan terbaik bagi proses injeksi tersebut. Kaedah cuba jaya adalah kaedah tradisional yang diguna pakai untuk menentukan keadaan terbaik tersebut namun ianya memerlukan masa dan tenaga yang banyak di samping meningkatkan kos pembuatan. Nilai dan keputusan yang dihasilkan oleh simulasi perisian dan injeksi sebenar berkemungkinan mempunyai sedikit perbezaan yang disebabkan oleh beberapa faktor dan faktor-faktor tersebut boleh dikenal pasti dengan melakukan kedua-dua analisa tersebut dan hasil analisa tersebut dibandingkan. Perbandingan tersebut akan menghasilkan peratusan kesilapan antara perisian simulasi dan injeksi sebenar.

TABLE OF CONTENTS

APPENDICES

LIST OF TABLES

Table No. **Page**

LIST OF FIGURES

LIST OF ABBREVIATIONS

CAE	Computer Aided Engineer
CAM	Computer Aided Manufacturing
FEA	Finite Element Analysis
MPA	Moldflow Plastic Advisor
ABS	Acrylonitrile-Butadiene-Styrene
MIMO	Multiple-Input Multiple-Output
MPI	Moldflow Plastic Insight

CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

Nowadays, plastic injection molding is become one of the important industry in the world. This industry placed in the manufacturing field and most of the parts, objects and goods surrounding are based on plastic material. Injection molding is one of the manufacturing techniques from manufacturing engineering field in producing parts that based on plastic material. The molten plastic that has been melted from the hopper through the barrel will be injected at high pressure into a mold with the cavity of desired parts shape. Major problem in plastic injection molding industry is the results are somehow different from the simulation software. Thus, it will contribute serious problems for the Quality Assistant and the engineer in order to predict the suitable setting or design of the part and mold. This project will compare those results. The parameters need to be selected as not all of the data or results can be observed by naked eye by actual injection. Published software, Moldflow will be used during analysis by simulation. In the end the results gained from those two approaches will be compared and analyzed to observe if the results are same, acceptable or errors.

This analysis can be done in many approaches but based on this project's title the study has to be done by manual experimental and at the same time by plastic simulation software. Benchmarking can be defined as comparison or to differentiate two or more parameters that have been studied. From the title of this project, in other words, it means to make comparison of the results, observations and consequences between the actual manually handled plastic injection molding machine and the results that have been analyzed by the plastic simulation software. There are some analysis can be compared between the actual injection and software simulation. For an example is the common defect occurred in plastic manufacturing industry which is shrinkage. Volumetric shrinkage is the contraction of polymer due to the change in temperature from melt temperature to ambient temperature [1]. High volumetric shrinkage can cause part warpage, sink marks, critical dimensions that are too small and internal voids. Excessive wall thickness and inadequate packing can both contribute to high volumetric shrinkage in a part. The solutions to avoid this defect are altering the part design such as the wall thickness and the other critical area. Second solution is altering the gate locations and lastly altering the processing conditions by increase the packing pressure. There other analysis that can be benchmark is the deflection of the finished products. The deflection resulting from the Moldflow shows the deflection at each node of the part (warpage or stress analysis), or each node of the wire or paddle (microchip encapsulation analysis), based on a "best fit" technique, where the original geometry and the deformed geometry are overlaid in such a way that they best fit together, or based on a defined anchor plane defined. There are a number of possible variants of the deflection result according to:

- Analysis type - The result name may indicate the type of analysis that was run, that is, either small deflection or large deflection. If this is not indicated in the result name, then the results will apply to a small deflection analysis.
- Net vs component deflections - The net view of net deflections at each node, or the component of the deflection either along the X, Y or Z axis. The axis directions are determined by the defined anchor plane and are indicated in the anchor plane symbols.

- All effects versus warpage contributors - There are four sets of deflection results. To create these results, run a small deflection warpage analysis and select the Isolate cause of warpage option on the Warp Settings page of the Process Settings Wizard.

There are also analyses that can not be compared between those two approaches yet it is important to be analyzed such as for an example the fill time analysis. As in Moldflow software the results of this analysis is called fill time result. The Fill time result shows the position of the flow front at regular intervals as the cavity fills. Each color contour represents the parts of the mold which were being filled at the same time. At the start of injection, the result is dark blue, and the last places to fill are red. If the part is a short shot, the section which did not fill has no color. Secondly, the analysis of time to freeze also an important parameter yet can be observed by naked human eyes. Thus, from Moldflow judgments the Time to freeze result is generated from a Midplane and Fusion flow analysis, and shows the amount of time taken from the end of fill at 100% to the ejection temperature. This result takes into account the dynamics of both filling and packing phases, where new hot material enters the cavity. This new hot material affects the cooling time.

Shrinkage is the amount by which a molded product is smaller than the size of cavity space wherein it was produced by injecting plastic under high pressure injection and at high temperature [2]. There are three rules regarding the shrinkage behavior which the first rule is, there is a definite relationship between pressure (P), volume (V) and temperature (T). This relationship is different for various plastic. Any and all conditions that affect those parameters will affect the shrinkage. Second rule is when a volume of plastic is heated it will expands. Then when it cools to the original temperature, it will contract to the original volume. Third rule is when a plastic is compressed the volume will be reduced. When the pressure is reduced to the original pressure it will return to its original volume. The greater the temperature difference between the room temperature and injected plastic then the shrinkage also will be greater. Timing also can affect the shrinkage behavior where the longer the injection pressure is kept on the plastic in the cavities the less will be shrinkage. In term of pressure, where the pressure on the plastic (in cavity) is high, less shrinkage will take place but in the other hand when the area is low in pressure the plastic will shrink more. It also can be affected by plastic material characteristics. Each plastic has a typical coefficient of temperature expansion. In most cases it is impossible to predict with certainty the correct shrinkage of a material since it depends on so many factors.

1.2 PROBLEM STATEMENT

1. The differences of the result between the software and the machine are not 100% the same
2. The detail about the defects that can not be analyzed by using simple observation methods have to be determined by using software simulations
3. The condition of mold and software capabilities might influence the results of both analyses.

1.3 PROJECT OBJECTIVES

a. To get the parameter value using the CAE or FEA software – by Moldflow software
b. To use the data from Moldflow to setup plastic injection molding machines
c. Determine the differences of results of the parameters between actual experiments and software simulation analysis

1.4 PROJECT SCOPES

a. Literature review will be done regarding to the title of this project
b. For this project Moldflow Plastic Advisor (MPA) software will be used for the software analysis method.
c. Reversed engineering will be applied according to the already available mold to obtain the parameters of the mold
d. The product designation is depends on the finished product and for this project the product is paper rack.
e. The material type will be used is Acrylonitrile-butadiene-styrene (ABS).
f. The machine that will be used is Arburg 520C Allrounder 2000-800 for the actual injection analysis.
g. The processing properties of the material will be used as reference for the software analysis
h. The model of the product is design by using CAE software Solidwork
i. The machine will be setup by using results from the software for the actual injection analysis.
j. The result from software analysis and the result from actual injection analysis will be compared.

CHAPTER 2

LITERATURE REVIEW

2.1 INTRODUCTION

Parts that based on the plastic material can be produced in many ways and the most popular approaches are by injection. The basic concept of this method is injecting the molten plastic material into the mold with the cavity of the product's shape and the material will be cooled down then forming into solid form of the desired product's shape before it ejected by pin ejector and ready to be used. In injection molding process, there are four main steps or cycles have to be taken namely filling cycle, cooling cycle, mold open cycle and part ejects. The crucial step is during the filling cycle since the quality of the goods and lifespan of the molds are depending seriously on this.

2.2 INJECTION MOLDING MACHINE

Basically, the injection molding machine functions as the holder of the mold and injecting the molten material into the cavity inside of the mold. There are several types of injection machine but most widely used are hydraulic type, all-electric and combination of both types. Generally, an all electric type machine not very different from hydraulic type in term of body mechanism [3]. However, there are also significant differences between those two types of machine and the differences as stated below:

i. the uses of AC Servo Motor
ii. the uses of ball screw

iii. the uses of gear and timing belt

The existences of these components are to substitute original hydraulic element such as hydraulic motor, directional valve, hydraulic board and cylinder. Since the electric elements are used to drive the injection machine so it is therefore called "All-Electric". The advantages of this type of injection machine are no problem with oil leakage as it does not use oil for hydraulic system thus will generate less pollution. It also has less operation noise, less energy consumption and has high accuracy of mold movement. In the other hand the operation cost of the machine is high with high cost of servo motor. The durability of ball screw also needs to be put under consideration since it has certain lifespan. This machine has slight difficulty on developing large tonnage force model which can resulted instable power supply and also unable to use accumulator to create transient high pressure. An injection molding machine is called hydraulic type when it use hydraulic system to open or closed halves mold by a reversible fluid motor actuated by a die control valve. The advantages of hydraulic type machine are the mold is easier to be setup onto the machine, the clamp pressure can be easily determined, low maintenance cost with low platen deflection since the force concentrated at the center of the platen. Vice versa the disadvantages of this machine are the oil for the hydraulic system tends to leakage and it requires large volume of hydraulic oil. The energy consumption is inefficient and overcompensate is a must due to compressibility of the oil. This machine also required large space.

2.3 IMPORTANT COMPONENT IN PLASTIC INJECTION MOLDING MACHINE

Plastic injection molding machine consists of several components that assembled into a whole machine.

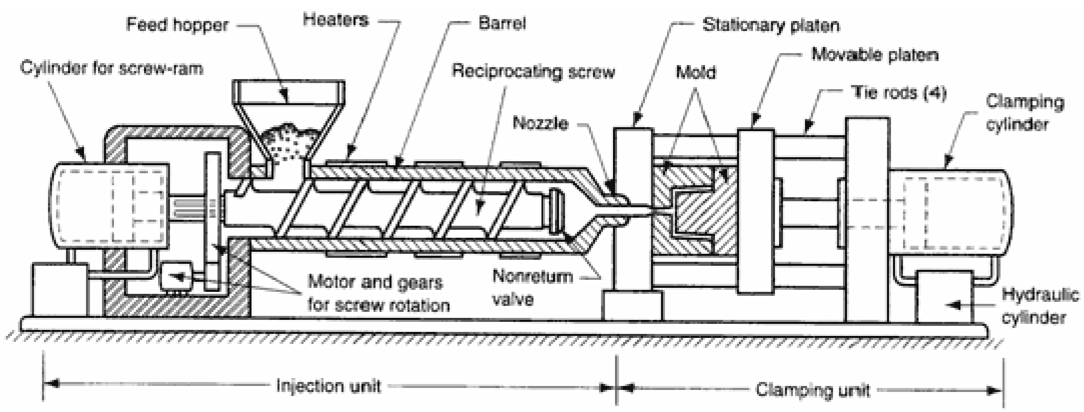

Fig 2.1: Important part of injection molding machine
Source: Plastic Technology, BMF 4713 Teaching Handout (2008)

As referring to the diagram, there are two main unit in the injection molding machines where they are stated as injection unit and clamping unit. In the clamping unit is consisted by stationary platen, mold, moveable platen, tie rods, clamping cylinder and in this case hydraulic cylinder since the machine is hydraulic type. The function of clamping unit is to holds the mold together, open and closed the mold automatically, and finish the injection process by ejects the finished product.

2.4 MOLD

Mold can be divided into two main types which are two-plate mold and three-plate mold type. The main difference of these two types of the mold is about the function of handling the runner. Three-plate mold has self-degating function which means the runner is disassembled from the finished injected products by mean of mechanical movements of the assembled mold. The three-plate mold has an extra plate compared to the two-plate mold with present of stripper plate assembled between the fixed half mold plate and top plate of sprue bushing. This function will produce two parting line instead of a single parting line for the two-plate mold type where the extra parting line located at the fixed half mold plate and stripper plate.

Three-plate mold is better when the quality of the surface finish on the products is crucial matter since the runner and sprue part do not have to be cut manually by manpower which the quality of the cutting will not be consistent with extra cost consumed for the salary man. The detached sprue and runner will be treated as wastes and depending on the material it can be recycle by crushing them back into particle or pallet form. If the material used categorized in thermoset family it can not be recycled since the chemical degradation of the material will be not resulting into a desired finished product and can be harmful for the screw where it can burning inside the barrel.

2.5 RUNNER

Runner is channel into the mould plate to connect the sprue and gate to impression. The type of runner can be defined as one of the most important factors that should be considered before fabricating process and mainly there are two types of runner namely cold runner and hot runner. They can be known by present of filament at the runner where hot runner type is chosen for one mold. There are some significant criteria differences of the two types of runner. The cold runner system has some disadvantages such as high cost of energy and workmanship, high scrap ratio, low product quality of surface appearances and requirement of high injection pressure. In the other way, hot runner system is able to provide precisely adjustable process temperature, uniform filling in multi-cavity molds, even heat distribution in the molds, improvement on mechanical properties of the injected products, cuts in production cost and shorter mold opening distance because absence of sprue while shorten the cycle time. The layout of the runner system also needs to be considered as critical factor which it depend on the shape of desired product and size. There are four main layouts such as conventional

[Fig 2.2], improved [Fig 2.3], balanced H [Fig 2.4] and circular [Fig 2.5]. When designing a mold the criteria of the mold is need to be categorized into consideration. The runner should be providing maximum cross sectional area from the standpoint of pressure transfer and a minimum contact area from the standpoint of heat transfer.

The following factors are should be considered while deciding the runner size. The first factor is about the wall section and volume of the molding. The cross sectional area of the runner must be sufficient to permit the molten material to pass through and fill the impression before the runner freezes. The second factor is the distance between impression and main runner or sprue where the resistance of flow is greatly depends on the length of the runner. When the gap between the impression and sprue or main runner is large it will make larger resistance for the flowing molten material. Thirdly is about the runner cooling system where the large size of runner will increase the cooling time.

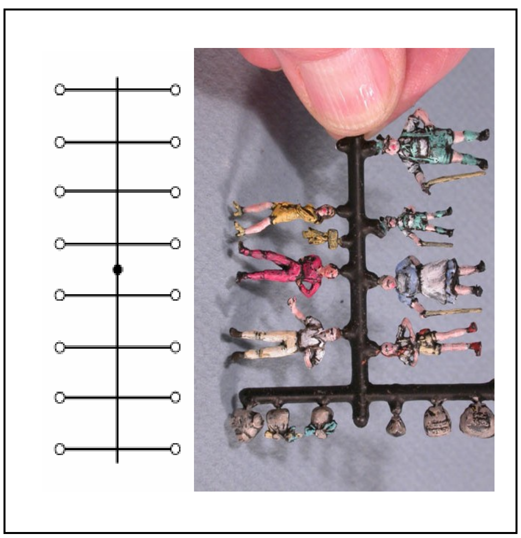

Fig 2.2: Conventional layout

Source: [3]

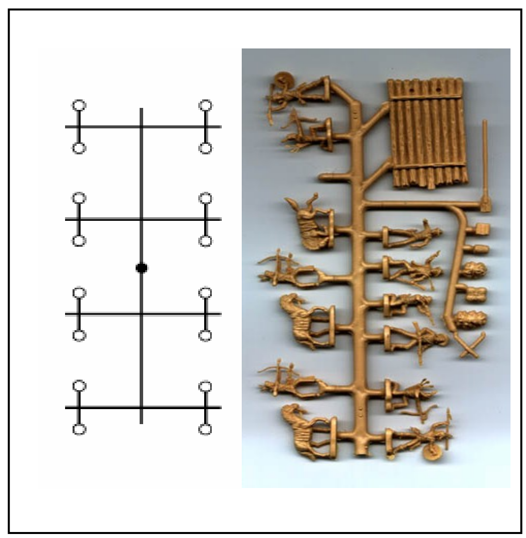

Fig 2.3: Improved layout

Source: [3]

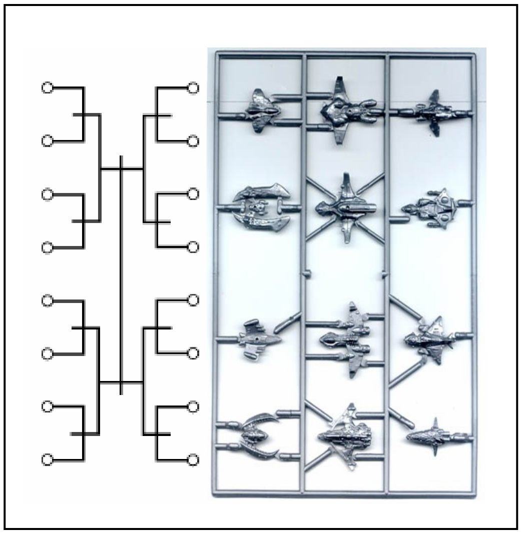

Fig 2.4: Balanced H layout
Source: [3]

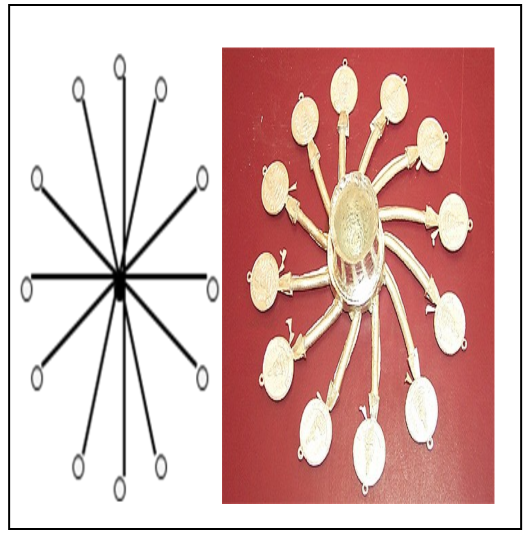

Fig 2.5: Circular layout
Source: [3]

2.6 DEFECTS ON PLASTIC INJECTION MOLDING

The defects such as burn marks or air burn which is brown or black burnt areas on the part located at furthest points from the gate is because of the tool lacks venting or the injection speed is too high [4]. The other type of defect is flash or burrs can be detected on the part when excess material in thin layer exceeding normal part geometry resulting from too much injection speed or too much material injected, clamping force too low or tool damaged. Sink marks can be detected as localized depression which happened at thicker zone of product. This defect occurred when the holding time or the pressure too low, cooling time too low with sprueless hot runners and this defect also can be caused by the gate temperature being set too high. The other type of defect is short shot where the finished product is only partial of the original shape. This is because lack of material, injection speed or pressure too low. Warping or also defined as twisting is when the part is distorted due to cool time is too short, material is too hot, lack of cooling around the tool or incorrect water temperatures (the parts bow inwards towards the cool side of the tool). Weld line or meld line is detected as discolored line where two flow fronts meet. The defect because of the mold or material temperatures set too low which mean the material is cold when they meet so they don't bond uniformly.